WL-TR-94-3127

Guide to ASIAC Computer Programs

WRIGHT LABORATORY

Gordon R. Negaard

CSA ENGINEERING
2850 W. BAYSHORE ROAD
PALO ALTO CA 94303-3843

July 1994

FINAL REPORT FOR PERIOD FEBRUARY 1991 - JANUARY 1994

FLIGHT DYNAMICS DIRECTORATE
WRIGHT LABORATORY
AIR FORCE MATERIEL COMMAND
WRIGHT PATTERSON AFB OH 45433-7542

NOTICE

When Government drawings, specifications, or other data are used for any purpose other than in connection with a definitely Government-related procurement, the United States Government thereby incurs no responsibility or any obligation whatsoever. The fact that the Government may have formulated or in any way supplied the said drawings, specifications, or other data, is not to be regarded by implication, or otherwise as in any manner, as licensing the holder or any other person or corporation; or as conveying any rights or permission to manufacture, use, or sell any patented invention that may in any way be related thereto.

This report is releasable to the National Technical Information Service (NTIS). At NTIS, it will be available to the general public, including foreign nations.

This technical report has been reviewed and is approved for publication.

CHRISTOPHER E. WHITE, 1st Lt, USAF
Project Engineer
Design & Methods Development Section
Design Development Branch

NELSON D. WOLF
Technical Manager
Design & Methods Development Section
Design Development Branch

GEORGE R. HOLDERBY
Chief
Design Development Branch
Structures Division

If your address has changed, if you wish to be removed from our mailing list, or if the addressee is no longer employed by your organization, please notify WL/FIBAD Bldg 45, 2130 Eighth St Ste 1, Wright-Patterson AFB OH 45433-7542 to help us maintain a current mailing list.

Copies of this report should not be returned unless return is required by security considerations, contractual obligations, or notice on a specific document.

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. AGENCY USE ONLY *(Leave blank)*	2. REPORT DATE	3. REPORT TYPE AND DATES COVERED
	July 1994	Final 02/01/91-01/31/94

4. TITLE AND SUBTITLE
Guide to ASIAC Computer Programs

5. FUNDING NUMBERS
C F33615-90-C-3211
PE 62201
PR 2401
TA 02
WU 94

6. AUTHOR(S)
Gordon R. Negaard

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
CSA Engineering, Inc.
2850 W. Bayshore Road
Palo Alto, CA 94303

8. PERFORMING ORGANIZATION REPORT NUMBER

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
Flight Dynamics Directorate
Wright Laboratory
Air Force Materiel Command
Wright Patterson AFB, OH 45433-7542

10. SPONSORING/MONITORING AGENCY REPORT NUMBER
WL-TR-94-3127

11. SUPPLEMENTARY NOTES

12a. DISTRIBUTION/AVAILABILITY STATEMENT
Approved for public release;
distribution is unlimited

12b. DISTRIBUTION CODE

13. ABSTRACT *(Maximum 200 words)*

ASIAC is an on-site contractor to the Structures Division of the Wright Laboratory's Flight Dynamics Directorate. In analyzing and solving aerospace structures technical problems, ASIAC utilizes approximately ninety structural analysis computer codes.

The computer programs range from small codes, designed for special purposes, to large general purpose structural mechanics codes. These computer programs can be useful tools to a designer or manufacturer to use to calculate internal mechanical and thermal stresses, predict structural strength, stiffness, weight, fatigue life, or to design for minimum weight and maximum life.

This handbook discusses the capabilities of each program and provides guidance in selecting a code to meet an individual need.

14. SUBJECT TERMS Computer Programs, Structural Analysis, Optimization, Fracture, Crack Propagation, Life Prediction, Structural Integrity

15. NUMBER OF PAGES 33

16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
Unclassified	Unclassified	Unclassified	UL

NSN 7540-01-280-5500

Standard Form 298 (Rev. 2-89)
Prescribed by ANSI Std. Z39-18
298-102

Contents

Tables

FOREWORD

This report was prepared by the Aerospace Structures Information and Analysis Center (ASIAC), which is operated by CSA Engineering, Inc. under contract number F33615-90-C-3211 for the Flight Dynamics Directorate, Wright-Patterson Air Force Base, Ohio. This work was performed under the ASIAC basic operation.

1.0 Introduction

ASIAC is an on-site contractor to the Structures Division of the Wright Laboratory's Flight Dynamics Directorate. In analyzing and solving aerospace structures technical problems, ASIAC utilizes approximately ninety structural analysis computer codes. One of the many services provided by ASIAC is making these programs available to qualified users, primarily to other government agencies or aerospace companies for a nominal fee to cover mailing costs. Most of these codes were developed under DoD sponsorship and are available to all DoD organizations without any restrictions. These programs are also provided to universities and to industry in most cases. However, these programs are provided only after the requester signs the necessary "Statement of Terms and Conditions" provided by ASIAC. The purpose of this step is to assure that the user (requester) fully understands that neither the Air Force nor any of its contractors involved in the development of the subject programs is liable for use/(misuse) or errors in the programs. In addition, the programs are provided for the use of the user and the user's organization only, and no secondary distribution is permitted.

The computer programs range from small codes, designed for special purposes, to large general purpose structural mechanics codes. These computer programs can be useful tools to a designer or manufacturer to use to calculate internal mechanical and thermal stresses, predict structural strength, stiffness, weight, fatigue life, or to design for minimum weight and maximum life. The purpose of this handbook is to discuss the capabilities of each program and to help provide guidance in selecting a code to meet an individual need.

To obtain additional information on the availability of programs and current service charges, contact ASIAC at:

WL/FIBAD/ASIAC Bldg 45
2130 Eighth St Ste 1
Wright-Patterson AFB OH 45433-7542

(513) 255-6688 DSN 785-6688
Internet asiac@fltvc1.flight.wpafb.af.mil

2.0 Background

Most older structural analysis codes were written for use on large, mainframe computers such as CDC, VAX, IBM, or even CRAY, and originally required experienced computer specialists to install and use the analysis programs. These programs often contained machine dependent language for some subroutines in order to utilize available memory most efficiently and to operate faster. As computers became faster and cheaper, most programs came to be written entirely in some version of FORTRAN.

All the present ASIAC codes are in FORTRAN abut many were written to run on older mainframes and must be ported to new mainframes and workstations. Inexpensive microcomputers are now reaching the speed and capacity sufficient to run many computer codes and can do it in a more user friendly environment than has been traditional on older, larger mainframe computers. In addition to the lower cost, another incentive is the versatility of graphic displays in the PC and workstation environment. Some of the latest ASIAC codes were created especially for the PC or have PC versions.

3.0 Description of Computer Codes

The discipline of solid mechanics can be divided into a number of subdisciplines depending upon the material properties of a structure, the forces acting upon it, and the characteristics of the deformation which is to be analyzed. One set of such categories was proposed by David Bushnell, Lockheed Palo Alto Research Laboratory, as follows:

1. Linear statics
2. Nonlinear statics
3. Linear dynamics
4. Nonlinear dynamics
5. Shock and vibration
6. Stability and buckling
7. Optimization
8. Thermal stress and creep
9. Viscoelasticity
10. Plasticity
11. Fracture or crack growth
12. Composites
13. Seismic Interaction
14. Welding

Structural analysis computer codes have been developed to analyze and predict the response of structures in all of these categories. ASIAC has collected a wide variety of codes capable of solving problems in almost all the above subdisciplines. For convenience, ASIAC groups the codes into eight functional categories according the purpose and capability of each code.

1. Linear structural analysis. This category includes programs that will perform linear static or dynamic analysis, or both. The programs are based on materials which respond linearly as a function of loading or displacement. The programs usually include the ability to predict thermal as well as mechanical stresses.

2. Nonlinear structural analysis. This category includes programs that will perform nonlinear static and/or dynamic analysis. Nonlinear effects may be from materials which respond nonlinearly or may be due to large displacements or deformations. Some programs also include the effects of thermal creep, viscoelasticity and plasticity. Buckling may also be included.

3. Layered composite analysis. This is a category of specialized analysis for calculating the micromechanical behavior of advanced composite materials such as multilayered fiber composites or honeycomb sandwich materials. They are generally used to assess the ability of a composite to carry structural loads or to create a design capable of carrying a prescribed load.

4. Structural joint analysis. This category consists of codes used to analyze bolted or bonded joints in both metal and composite structures. They are generally used to predict joint strength or fatigue life.

5. Structural integrity analysis. This is a category of programs based on fracture mechanics, which can be used to predict fatigue life or catastrophic failure of a structural part. These are generally used to predict the effects of notches or cracks on structural performance.

6. Structural optimization. This is a category of programs usually used to optimize a structure or component for minimum weight. A number of specialized codes are available to optimize such items as girders, columns, transmission towers, or stiffened panels, in addition to optimizing built up structures such as an aircraft wing.

7. Auxiliary programs. This is a category of helpful programs that are typically used as pre and post processors for analysis programs. They perform functions such as converting data for use in one code to a format that is used in another, or to provide information for debugging or improving a structural model. Such programs are also often used to plot both deformed and undeformed structure or graphically display stresses, temperatures, and deformations.

8. Miscellaneous programs. This is a category used for programs that do not fit conveniently into one first seven categories. Examples are programs for calculating blast or projectile damage to structures, hydraulic ram effects for explosions or projectiles entering fuel tanks, or for thermal problems involving radiation in enclosures.

4.0 Listing of Computer Codes

A complete listing of structural analysis computer programs currently supported by ASIAC follows.

4.1 Linear Structural Analysis Programs

ANALYZE

This is a small finite element program useful for static analysis of problems containing 150 to 200 degrees of freedom and a comparable number of elements. The program library consists of a bar, membrane triangle and quadrilateral, and a shear panel. This program is extremely useful in training engineers in the use of finite element programs, in the development of finite element models of aerospace structures and in research in structural analysis and optimization. Last updated in 1978.

ANALYZE-PC

This is a micro-computer version of both ANALYZE and DANALYZE. The program is written in Microsoft FORTRAN for use on Z-100 and Z-150 micro-computers, but it will run on most IBM-PC compatible machines. Last updated in 1986.

ASAAS

This is a version of SAAS III extended to solve problems which involve asymmetric mechanical and thermal loads and orthotopic materials with asymmetric temperature dependent mechanical properties. All dependent variables, including the mechanical properties, are represented by a Fourier series expansion circumferential coordinate. Since only a limited number of harmonics are allowed, the asymmetric loading must be fairly smooth. Last updated in 1971.

BARANALYZE

This program is a smaller version of the ANALYZE code and focuses on the analysis of frames using bar elements. The source code is available in FORTRAN for an IBM/XT and compatible using DOS 3.1 or higher. Last updated in 1987.

CRIP

This package consists of four computer codes for the design and analysis of curved metal and composite panels designed to operate in the postbuckling range. It includes two design programs, CRIP for compression panels and TENWEB for shear panels, and two analysis programs, COMPAN for compression panels and SHRPAN1 for shear panels. These programs are designed for interactive use and all inputs are conversationally solicited. Last updated in 1985.

DANALYZE

This is a small finite element program derived from ANALYZE. It is useful for in-house studies of the dynamic response of structures. The program uses a Sturm sequence property with a bisection procedure to bracket eigenvalues within a specified tolerance. Inverse iteration is used to compute eigenvectors and a Raleigh Quotient procedure refines eigenvalues. Last updated in 1982.

PLATE

This is a small finite element program for the elastostatic analysis of anisotropic plates of arbitrary geometry subjected to inplane, transverse, or thermal loads. Plate thickness need not be constant from element to element and initial stresses may be present. Last updated in 1974.

PLSTR

This is a small finite element program which performs an elastic, plane stress structural analysis. It can handle varying thermal and pressure loads. The program library consists of quadrilateral, triangle, and rod elements. A mesh generator included in the program is useful in reducing the required number of physical input cards when many orderly, similarly sized elements occur in the grid. Last updated in 1973.

SAAS III

This is a moderate sized finite element code for static analysis of axisymmetric solids using a quadrilateral ring element with linear displacement fields. SAAS III also includes an option for plane stress/strain bodies. The program accepts orthotropic temperature dependent material properties. The bodies can be subjected to arbitrary axisymmetric mechanical, thermal, and pore pressure loading. The mechanical loads can be surface pressures, surface shears, and nodal point loads as well as acceleration or angular velocity. Last updated in 1971.

SAP IV

This is a moderate sized finite element structural analysis program for static, free vibration, transient, or spectral response of large three-dimensional systems. The structural systems to be analyzed may be composed of combinations of nine different structural elements, including isoparametric membranes and solids. Especially recommended for seismic analysis. Last updated in 1974.

4.2 Nonlinear Structural Analysis Programs

NEPSAP

This is a large three-dimensional finite element program for the nonlinear thermoelastic-plastic and creep analysis of arbitrary three-dimensional structures undergoing large deformations. The library uses truss, beam, membrane plate bending, and isoparametric solid elements. Last updated in 1974.

NONSAP

This is a moderate sized finite element program which solves static and dynamic, linear and nonlinear problems. The nonlinearities may be due to material nonlinearity, in which case elastic, hyperelastic, and hypoelastic material behavior may be considered, or the nonlinear effects may arise from large displacements and large strains. The program library consists of a truss element, two-dimensional isoparametric solids with 4 to 8 nodes for plane stress, plane strain, and axisymmetric deformation, and three-dimensional isoparametric solids with 8 to 20 nodes. Last updated in 1984.

SBS

This is a small finite element program for small strain analysis of plane structures in the strain hardening elastic-plastic range. Bar and constant strain isotropic plane stress triangles are used in the idealization. Up to ten types of materials can be used at one time. Three types of stress-strain laws and proportional increment loading are options. Last updated in 1968.

4.3 Layered Composite Analysis Programs

AC-3

This program is similar to SQ5 in performing a point stress analysis or calculating laminate flexural and extensional constants under inplane loads, moment and temperature effects. It can handle up to 100 layers, with ten materials and ten loading conditions. Gross laminate properties, stresses, strains, and margins of safety for each individual layer can be output. Last updated in 1983.

AC-5

This program calculates the general stability of a honeycomb sandwich panel with orthotropic core and face sheets under biaxial loading. AC-5 will handle panels under uniaxial compression, biaxial tension-compression, and biaxial compression-compression. Last updated in 1983.

AC-7

This program calculates the deflection, moments, and shears in a flat orthotropic honeycomb sandwich panel under pressure plus in-plane loads, given material and geometry parameters. The method is a series summation, and the series terms as well as the series sum are printed out to enable the user to monitor convergence. The in-plane loads can be compression-compression, tension-compression, or tension-tension. Last updated in 1983.

AC-10

This program determines the buckling strength and/or stress resultant and displacement pattern for rectangular and skewed anisotropic plates. The dynamic frequencies for the lower natural modes can also be found. Both in-plane and lateral loads may be applied. Boundary conditions include any combination of five, clamped, single, or elastic supports. In addition, spring supports may be placed at prescribed locations. Last updated in 1983.

AC-11

This program calculates the general buckling load for a flat honeycomb sandwich panel with orthotropic face sheets and core under in-plane shear. The program uses a modified eigenvalue solution which involves convergence of the geometric means of succeeding eigenvalues. The face sheet thicknesses are assumed to be equal. Last updated in 1983.

AC-50

This program calculates the initial, final and design in-plane strength for symmetric balance advanced composite laminates which can be composed of plies of different materials and thicknesses. Ply failure is determined by either a stress or strain failure criteria. AC-50 accounts for ply failure and continues to compute the strength of the laminate configuration with failed plies until the initial, final, and design allowables are reached. Last updated in 1983.

CLAPP

This is a program for the collapse load analysis for plates and panels. The program uses a finite-difference, energy based procedure to calculate bifurcation or collapse of circular cylindrical panels or rectangular plates made of composite laminates augmented by longitudinal stiffness. The program is also capable of handling rectangular cut-outs and initial geometric imperfections. Boundary conditions are restricted to zero force or zero displacement. Last updated in 1981.

CREPAIR

This is a computer program for the methodology to design repairs for composite skins subjected to biaxial or shear loads. This methodology was extended to include numerous axisymmetric cutouts, thermal loads and taper patch analysis. Theoretical solutions are confirmed by industry test results. Last updated in 1987.

NOLAST

This is a program for nonlinear failure analysis of composite laminates. It predicts the response of multi-directional laminates subjected to inplane loads by using piecewise cubic spline interpolation functions to represent basic stress-strain data. The ultimate cumulative response of the laminate is determined by the ply-wise application of an energy based failure criterion. Last updated in 1976.

PANBUCK

This program uses classical solutions for stability analysis to predict the critical elastic buckling loads and modes for rectangular laminated composite plates or honeycomb sandwich panels. It handles panels subjected to in-plane biaxial compression or shear loads applied parallel to the panel's axes of orthotropy. Edge conditions may be fully clamped, simply supported or a combination of the two. Last updated in 1970.

SQ5

This program performs a point stress analysis of an advanced composite laminate under in-plane loads, moments and temperature effects. It will handle up to 100 layers with arbitrary orientation. It derives gross laminate properties, stresses and strains. It will give the shear stress distribution across the laminate thickness if the shear resultants, Qx and Qy, are known. It also has the capability of obtaining a laminate interaction diagram based upon the maximum strain theory of failure. Last updated in 1983.

WL3017 (CTQ23)

This program is designed particularly for solving the stress free field in free-edge delamination specimens made from laminated composites. Material properties are assumed homogeneous, linear elastic orthotropic in each ply. Stress and displacement components are calculated both on the surface and in the interior of the specimen. There are versions of the program for a PC, mainframe, and super computer. Last updated in 1991.

WL3018

This is a finite element computer program for the analysis of laminated composite plates based on a consistent shear deformable theory. The program is capable of computing deflections and stresses in laminated plates subjected to nodal and distributed loads. It can also be applied to free-edge delamination specimens. The program is written to run on a Cray X-MP/28 Supercomputer, but a standard mainframe version is also available. Last updated in 1991.

WL3019 (CHYOU)

This is a finite element computer program for the implementation of Pagano's theory of composite laminated plates. The program will carry out finite element analysis of laminated plates under in-plane as well as transverse loading. The program is written to run on a Cray X-MP/28 Supercomputer, but a standard mainframe version is also available. Last updated in 1991.

WL3020 (AXO3D)

This is a finite element computer program for the analysis of a composite laminated structure with axisymmetric geometry and loading. Circumferential displacement is considered in addition to radial and axial displacement. The program provides an economic alternative to a fully three dimensional analysis of free-edge delamination coupons. The program can also solve a variety of problems including tubular angle-ply specimen, nozzles, etc. Available in a PC as well as a standard mainframe version. Last updated in 1991.

4.4 Structural Joint Analysis Programs

A4EI, A4EJ, A4EK

These three programs constitute a package for the analysis of joints in both metal and composite structures. Program A4EI is for the nonlinear analysis of adhesively bonded stepped-lap joints and doublers. Program A4EJ is for the analysis of multi-row mechanically fastened bolted joints. Nonlinear analysis of combined bonded and bolted joints can be done using A4EK. All three programs are based on theory of elasticity solutions modified to handle the associated discontinuities and nonlinearities. Last updated in 1981.

AC-20

This program analyzes a symmetric stepped lap joint composed of an orthotropic adherent. homogeneous isotropic adherent and an adhesive. The program uses a Hill-Von Mises interaction equation to predict the combined axial and shear loading which will cause failure in each lap. Last updated in 1983.

BJSFM (AC-71)

This program, Bolted Joint Stress Field Model, is a small code developed for strength analysis of isotropic materials or anisotropic laminates at individual fastener holes. Input data required are lamina mechanical properties, in-plane loadings, hole geometry and hole loading. Lamina stress and strain distribution for combined bearing and bypass loads are obtained by superposition. Laminate failure is predicted by comparing elastic stress distribution with any of five material failure criteria on a ply-by-ply basis. The program is designed to run interactively, but can also be run in a batch mode. Last updated in 1983.

BONJO, BONJOIG, BONJOIS

The BONJO series of computer programs covers the analysis of narrow, uniaxially loaded bonded lap joints. Each adherent may be either an isotropic or laminated anisotropic plate. The analyses include the effects of inter-laminar shear and normal stresses, and residual stresses caused by bonding at elevated temperatures. Two versions are available for the linear elastic analysis of joints. BONJOIG will analyze any general single lap or double lap bonded joint. BONJOIS will analyze single lap bonded joints in which the two adherents are identical. BONJO approximates nonlinear material properties by means of a linear, elastic-totally plastic, stress-strain relationship. Last updated in 1974.

GRUMFIT
This is a small finite element program for predicting both the linear and nonlinear stresses and deformations of mechanically fastened joints. The idealization uses a set of stacked parallel plates which transfer planar loads among themselves by means of transverse fasteners. Applications include the calculation of residual stresses, fastener bending and shear deformation, bearing stress, and the prediction of fatigue life of typical mechanically fastened joints. Last updated in 1970.

JOINT
This program consists of a collection of composite joint analysis routines interfaced with TEKTRONIX graphic capability. The program is designed to allow a nonspecialist to interactively design and analyze various types of bonded or bolted composite structural joints. Last updated in 1978.

JTSDL
This program performs a nonlinear analysis of single or double lap bonded joint subjected to static loads at room temperature. The joints are assumed to be in a state of plane strain. The adherents may be either orthotropic laminates or isotropic material of constant thickness. The adhesive is assumed to be isotropic with constant thickness. Normal stresses through the thickness and interlaminar shear are neglected and each laminate is assumed to be symmetrical. Last updated in 1973.

JTSTP
This program is similar to JTSDL, but is designed to analyze step lap joints with a large number of steps which may be of arbitrary geometry. Last updated in 1973.

SAMCJ
This program, Strength Analysis of Multi-fastened Composite Joints, performs a strength analysis for laminated and/or metallic plates bolted together by many fasteners. SAMCJ is an interactive one step analysis that computes load distribution among many fasteners and predicts joint failure load, failure mode and failure location. Predictions of joint strengths are fairly accurate and generally conservative. Last updated in 1986.

SASCJ

This program, Strength Analysis of Single Fastener Composite Joints, allows the user to interrogate an isolated fastener location, accounting for nonlinear joint behavior due to progressive (two-stage) ply failures. The analysis predicts load ply failures and delamination until the bolted plate cannot carry any additional applied load. Last updated in 1986.

SCAN

This program can be used for the analysis and design of bolted joints. The developed analytical procedures consider the interactions of width-to-diameter (W/D) and distance-to-diameter (e/D) geometry variables as well as joint design variables. The analysis involves a boundary collocation method together with anisotropic theory of elasticity and lamination plate theory. The resulting methodology permits ply-by-ply stress (strain) analysis and is combined with a ply-by-ply strength analysis. Last updated in 1985.

4.6 Structural Optimization Programs

ACI-120

This program determines a least-weight symmetric balanced configuration for an unstiffened composite laminate. The panel is assumed to be rectangular and simply supported on all four sides. In-plane and thermal gradient loads using either a "first failure" stress or a strain strength criterion can be handled. The stability criterion assumes that the panel is infinitely long in the direction of the maximum compressive axial load. Last updated in 1983.

AC-31

This program determines the lightest weight balanced laminate configurations for a laminate containing prescribed angle plies and undergoing multiple multi-axial in-plane loading. The program obtains an upper bound to the lightest weight solution. Last updated in 1983.

AC-32

This program optimizes the weight and thickness distribution of a balanced laminated composite plate with a fixed number of plies and orientations. The optimum thickness of each of the plies is found. Last updated in 1983.

AC-33

This program optimizes a balanced symmetric laminate with prescribed angle plies for elastic buckling under biaxial inplane loading. The optimum thickness of each ply is found. Last updated in 1983.

ADDRESS

This is a finite element program for the Automated Design of Damage Resistant Structures. An optimization criterion approach is used to design damage tolerant structures subject to stress, deflection, and frequency requirements. Damage conditions are treated in an integral manner in the resizing algorithm. An interactive reanalysis procedure is used to improve the efficiency of the static analyses that are needed as the optimization proceeds. For wing structures, size and location of damage are the driving features of the process. Criteria are presented to aid the analyst in determining the critical stress allowables. Last updated in 1981.

ASOP III

This is a large finite element program for the minimum weight design of structures subjected to strength and deflection constraints. It utilizes element average stresses for resizing purposes. It can accommodate composite laminates with up to six fiber directions. The element library consists of bars, beams, plane stress triangles and quadrilaterals, warped quadrilaterals, warped shear panels, and a hinged beam. The output contains data for final design, including nodal deflections, member gages, weights, stresses, strains, internal loads, critical stress ratios, and critical load conditions. Last updated in 1976.

ASTROS

This program provides multidisciplinary analysis and design capability for aerospace structures. The capability of the system includes structural analysis, both static and dynamic, aeroelastic analysis and automated design. A specifically designed data base and executive system were implemented to maximize the system's efficiency, flexibility and maintainability. This program is still in the development stage and thus is available by special request only.

OPTCOMP

This is a small finite element program which uses an interactive procedure based on optimality criteria to design a minimum weight structure with composite laminates. The response of the structure to the applied loads is obtained by finite element analysis. The program library consists of a constant strain triangle and quadrilateral shear panels and bars. Last updated in 1976.

OPTIM III

This is a small finite element weight optimization code. It is written so that most of the input data cards are compatible with NASTRAN. The element library consists of eight elements: axial force members, pure shear and shear web elements, triangular membrane plates in plane stress, quadrilateral membrane plates in plane stress, midpoint triangle membrane plates in plane stress, midpoint quadrilateral membrane plates in plane stress and tubular beam elements. Orthotropic material properties can be used with all membrane plate elements. Composite plate structures can be modeled by stacking orthotropic layers of these elements. Three types of failure criteria for composite plates are used in the program: the von Mises energy of distortion criterion, maximum stress theory and maximum failure theory. Last updated in 1978.

OPTSTAT

This is a small finite element program intended for the minimum weight optimization of aerospace structures modeled with membrane elements and subjected to static loads. Stress, displacements and element size are used as constraints. The program library contains membrane triangles and quads that can be isotropic, orthotropic or layered composites. Bar and shear panel elements can also be used but must be input with isotropic material properties. Last updated in 1979.

4.5 Structural Integrity Programs

AC-41

This program determines the ultimate strength of a laminate containing either a through-crack or a hole under a far-field load. A characteristic distance failure criteria is used to determine progressively the minimum load which would cause laminate failure at prescribed points. The criteria states that failure occurs at a point of high stress concentration when the laminate stress tensor intersects the laminate strength tensor at a distance perpendicular to the boundary radius. Last updated in 1983.

APES

This is a finite element program designed to directly calculate the mode I and II stress intensity factors for structures containing up to five crack tips under conditions of plane strain, plane stress, or axial symmetry. Twelve-node conventional and enriched (crack tip) isoparametric elements are utilized; no special treatment around crack tips is required. The high order of the elements results in high accuracy and great economy in data preparation, leading to use of the program in non fracture applications as well. KINGKONG and KINGKONGII are larger versions of APES with additional capabilities. CHECK is a CALCOMP graphical data checker and plotter. Last updated in 1981.

APES-PC

This is an IBM-PC compatible interactive version of APES. The program requires 640K RAM to operate. A formatted ASCII input file is required. An arithmetic co-processor is not necessary, but will reduce execution time by a factor of two. Plot files can be produced in HPGL format. Last updated in 1991.

CRACKS IV

This is an automated program for analyzing crack propagation in homogeneous materials for cyclic loaded structures. The program allows a choice of Walker, Foremen or Paris crack growth models. Provisions are made for surface flaws, through-the-thickness cracks and the transition from one to the other. Load interaction effects for two retardation models are also included. Last updated in 1983.

CRKGRO

This program is a two-dimensional crack-growth computer routine to perform detailed fatigue crack analysis on a cycle-by-cycle basis. An improved load interaction model accounts for both retardation and acceleration effects of spectrum loadings. The program contains a crack library which consists of 10 subroutines, each containing a specific stress intensity factor solution for a specific crack geometry. The program provides the option for counting the cycles for spectrum loadings through the range-pair counting routine built into the program. It also provides the option to perform parametric studies. Last updated in 1981.

DAMAGE

This is a fracture mechanics based life prediction program for hypersonic airframe structures subjected to combined mechanical loadings and thermal profiles. The program accounts for the effects of temperature on yield strength and fracture toughness, and includes the effect of sustained loads at elevated temperatures on crack growth rate. Last updated in 1989.

DAMGRO

This program has the capability of simultaneously analyzing the crack growth and crack initiations at the edge of a hole. Various geometrical configurations may be analyzed by the use of several subroutines. The program contains ten crack growth and six crack initiation routines. The crack growth is governed by the stress intensity factor at the edge of the crack while the crack initiation is governed by the strain energy density at the edge of a hole. Last updated in 1986.

LUGRO

This is an automated computer program used to predict the residual strength and fatigue crack growth behaviors of single through the thickness cracks and single corner cracks at attachment lugs. The program contains the following four basic elements: stress intensity factor solution, baseline crack growth rate relationship, applied load sequence, and spectrum load-interactive model. Last updated in 1984.

MMCLIFE

This program was developed to insure structural integrity of fiber reinforced metal-matrix composites when applied to aircraft structures. Analysis methods were developed based on fiber and matrix properties to predict notched and un-notched static strength, crack initiation, fatigue failure mode, life to failure, and residual strength. Last updated in 1987.

MODGRO

This is an interactive IBM-PC compatible program to calculate crack growth. The program is similar to CRKGRO in compatibility and operation. The input model or spectrum information can be created on a tape editor prior to running the program, or the data can be created when running the program. The remainder of the program is run in an interactive prompt or menu mode. Last updated in 1992.

ORD

This is a portable computer based expert system for structural repair of battle damaged aircraft analysis. Capabilities include composite laminate strength, bolted and bonded joint repair, and substructures repair. The software requires a minimum of a 80386 Computer, math coprocessor, EGA or VGA monitor, and 2 Meg of memory and up to 6 Meg of hard disk storage space. Last updated in 1991.

PACL

This is a FORTRAN code for the post buckled analysis of stiffened metal or composite panels subjected to combined compression and shear loads. The program can be used to predict the out-of-plane displacements, in-plane displacements, and the strain or stress distribution within the panels. Stiffener strains are also predicted. The program requires access to IMSL to run. Last updated in 1989.

PBUCKL

This program performs a failure analysis of curved or flat stiffened metal or composite panels loaded in uniaxial compression and/or stress operation in the post buckling range.
The program is designed for interactive use on a mainframe, and all inputs are interactively solicited. The program results are in the form of margins of safety for the various failure modes and the corresponding failure loads. Last updated in 1989.

SEAFAN

This program, Sequence Accountable Fatigue Analysis, calculates the cumulative damage of notched structural members subjected to arbitrary spectra. It incorporates a local stress-strain approach with a residual stress relaxation analysis in order to improve the sequence sensitivity of cumulative damage analysis. An example spectrum and resulting cumulative damage analysis are illustrated. Last updated in 1973.

TRANSPORT

This program is specifically tailored to develop random cycle-by-cycle, flight-by-flight loading sequences typical of transport type aircraft structures. However, any type spectrum can also be generated. The input data consists of loads exceedance spectra or data to calculate such spectrum by the program. The output is a valley-peak sequence of the loads spectrum which may be written onto magnetic tape for use in testing or other analysis. Last updated in 1978.

4.7 Auxiliary Programs

NASPECT

This is a NASTRAN preprocessor for determining aspect ratios of all two- and three-dimensional elements. It also determines if the quadrilateral elements or the sides of the three-dimensional elements are out of plane or warped. Warning messages for all bad elements are printed out. It determines the length of all one dimensional elements and checks the orientation of the inertia vector for BAR and BEAM elements. Also, the program converts all coordinates to the basic system, and these may be displayed or saved. Last updated in 1977.

NASSET

This is a NASTRAN preprocessor which determines the plot set membership of a model using qualifying statements. These qualifying statements reference the basic coordinate system. These statements may be complex and multiple (e.g.,$-3.0 < x < 6.5$, $y + z > 5.O$, 4 and $y > O$). These sets, which are output on the punch file, may be used directly in the NASTRAN Case Control Deck. Last updated in 1977.

NASSORT

This is a NASTRAN preprocessor for listing elements by type (QUAD, BAR, etc.) that are connected to each grid point. This information is very desirable for debugging and changing models. Last updated in 1977.

NASSAP

This is a preprocessor which takes a SAP IV input deck and translates it into a NASTRAN format. The program is internally commented. Last updated in 1986.

NASTIDY

This is a preprocessor used to resequence the grid numbers in a NASTRAN bulk data deck. The renumbering scheme may be based either on SEQGP cards or on the sorted order of the GRID cards and may start at any number desired. A new deck with corrected grid and element cards is output. The program is internally commented. Last updated in 1982.

TRANSNAS

This is a preprocessor which translates a NASTRAN bulk data deck into a SAP IV format. Last updated in 1976.

4.8 Miscellaneous Programs

BR-1FC

This is a large finite element program for predicting the structural response of fiber composite aircraft structures to HE projectile detonations. Deflections, stresses, strain, and element failures are calculated using a finite element analysis with a laminated orthotropic elastic-plastic capability. The user has the option of selecting the force as direct input data or letting the code compute the pressure by specifying the size and type of an explosive. Last updated in 1978.

BVDS

This program, **B**eam **V**iscoelastic **D**amped **S**tructure, can be used by designers to estimate damping in beam-like structures with viscoelastic constrained layer damping. The code consists of four general purpose computer programs which analyze and graphically present results in the form of carpet plots for multi-layered beam structures. The program was written for VAX/VMS operation and requires DI3000 software for plotting. The user can plot modal loss factor, modal frequencies, RMS response, and peak resonance, each as a function of temperature. The program can output the data in tabular form which could be used on other plotters. Last updated in 1990.

HRSR

This program allows the prediction of hydraulic ram effects on fuel tanks made up of multi-layer symmetric laminates. The program calculates hydraulic ram pressures, deflections, strains and failures. The transient stresses and strains in individual plies of the finite elements are computed and printed. HRSR provides a useful method to predict the general nature of the hydraulic ram structural response, particularly for preliminary design studies where the relative merits of alternate design configurations are being evaluated. Last updated in 1985.

IHCP2D

This is a program for the solution of general two-dimensional Inverse Heat Conduction problems. The program is used to estimate an unknown surface flux from distribution by utilizing transient temperature measurements within the body. It uses the TOPAZ heat conduction code to obtain the direct heat conduction solution and then produces the inverse solution by exact matching of the measured and calculated internal temperatures. Last updated in 1989.

INBLAST

This program predicts the blast inside a closed or vented structure. It calculates the combined the combined shock wave and pressure time-history for explosions. Structural shapes are limited to parallelepipeds, L-shaped rooms, and cylinders with regular and irregular polygonal cross-sections. The program is designed to run on any IBM-AT or newer desktop computer with a math co-processor. Last updated in 1991.

SINDA

This is a large numerical differencing program for the solution of transient thermal problems using a lumped parameter network representative of the physical problem. The program contains subroutines for handling interrelated complex phenomena such as sublimation, diffuse radiation within enclosures, and simultaneous 1-D incompressible fluid flow, including valving and transport delay effects. Last updated in 1979.

TM 5-1300

This is a computerized edition of the Tri-Service Structural Design Manual ''Structures to Resist the Effects of Accidental Explosions,'' published by the Army as TM 5-1300, by the Air Force as AFR 82-22, and by the Navy as NAVFAC P-397. The manual is approved for blast-resistant designs in explosive safety applications. This manual should be used for all new protective construction applications intended to satisfy DoD explosives safety criteria. The program runs on any IBM compatible desktop computer. Last updated in 1991.

5.0 Program Selection

It is not the intention of ASIAC to compete with the providers of commercial software or to distribute programs available from software centers such as COSMIC. ASIAC programs are usually not general purpose production codes. If a user needs a large scale production type analysis capability, one of the better known commercial general purpose programs, such as ABAQUS, ADINA, ANSYS, MARC, NASTRAN, or NISA, is probably the better choice. However, these programs are quite expensive, running in the tens of thousand of dollars per year. General purpose microcomputer based programs such as ALGOR or COSMOS-M are becoming competitive in capabilities, size, and speed, usually at only a fraction of the cost.

The programs that ASIAC uses and maintains are directed toward research and development purposes such as the development of new analysis techniques or codes, or for special analysis purposes or for teaching. ASTROS, a code still in the development phase, although meant as a research tool, is the closest to a multi-purpose analysis tool of any of the programs in the ASIAC library. NEPSAP and NONSAP, while relatively old and primitive nonlinear programs, can still be useful in a production mode to a user with a limited budget. With the principal exception of ASTROS, the codes maintained by ASIAC do not require an extensive learning curve or training courses. Subroutines can also be extracted from these programs for use in other applications.

Selecting an appropriate analysis program for a particular purpose can be difficult since there are so many choices available. The abstracts in Section 4 are intended to contain enough information about each available code to ensure the selection of the appropriate code for a user's purpose. The following tables are included to help narrow the field of choices and make such a selection easier. The tables are meant to highlight the major elements of each code and make comparisons easier.

Table 1 provides a quick summary of the ASIAC finite element structural analysis codes, both linear and nonlinear. The table lists the analysis capability, and the elements, materials, and loading available in each code. There is not a standardized nomenclature for many elements. The nomenclature used in this table is meant to be as descriptive as possible. The truss element is defined as a simple tension or compression element with no bending capability, corresponding to the rod element in NASTRAN, although some codes refer to this as a beam element. The beam element in this table is an element which will carry a bending moment and corresponds to the NASTRAN bar element. The table also gives the results that can be stored or printed from the output.

Table 2 provides a condensed summary of the layered composite codes maintained by ASIAC. The table lists the analysis capabilities contained in each code, the input necessary to the code, the type loading that each code can utilize, and the output provided by the code.

Table 3 summarizes the capabilities of codes available for analyzing or designing structural joints. The types of joints that each code can handle are itemized, as well as the input necessary to the code, the type loading used in the code, and the output provided by the code.

Table 4 summarizes the optimization code capabilities. Some of these codes are finite element codes capable of handling structures. Others are useful only for designing individual panels or laminates. Element libraries are listed for the finite element codes. Again, input and loading choices are listed, as well as the output provided by each code.

Table 5 summarizes the capabilities of the different fracture mechanics and structural integrity programs available. The program capabilities are itemized. The input and loading tables summarize the types of information necessary to utilize the code. The output available to the user is also summarized.

Table 1. Structural Analysis Codes

Capabilities \ Codes	ANALYZE	ASAAS	CRIP	DANALYZE	PLATE	PLSTR	SAAS III	SAPIV	NEPSAP	NONSAP	SBS
Procedure Library											
Static Analysis	●	●			●	●	●	●	●	●	●
Modal Analysis								●		●	
Transient Dynamic Analysis				●				●			
Buckling Load Determination			●								
Post Buckling Analysis			●								
Thermal Analysis		●			●	●	●		●		
Element Library											
Truss	●			●		●		●		●	●
Beam								●	●		
2D Solid							●		●	●	●
3D Solid								●	●	●	
Axisymmetric Solid		●					●	●	●	●	
Shear Panel	●			●							
Membrane	●			●		●			●	●	●
Plate Bending					●				●		
Thin Shell								●	●		
Thick Shell								●	●	●	
Axisymmetric Shell								●	●		
Material Library											
Linear Elastic Isotropic	●	●	●	●	●	●	●	●	●	●	●
Linear Elastic Anisotropic		●	●		●	●	●		●	●	
Nonlinear Elastic									●	●	
Plasticity									●	●	●
Temperature Dependent Properties									●		
Loading											
Concentrated Loads	●	●		●	●	●	●		●	●	●
Pressure Loads		●	●		●	●	●	●	●	●	
Body Loads					●	●	●	●			
Nodal Temperatures		●			●	●	●	●			
Prescribed Displacement								●	●		●
Initial Stress / Strain										●	
Output											
Element Stresses	●	●			●	●	●		●	●	●
Element Strains		●				●	●				●
Element Strain Energy	●										
Nodal displacements	●	●	●		●		●		●		●
Nodal Forces	●								●		●
Eigenvalues				●				●		●	
Eigenvectors				●				●		●	
Margins of Safety	●		●								

Table 2. Layered Composite Analysis Codes

Capabilities / Codes	AC-3	AC-5	AC-7	AC-10	AC-11	AC-50	CLAPP	CREPAIR	NOLAST	PANBUCK	SQ-5	WL3017	WL3018	WL3019	WL3020
Program Capability															
Laminate Strength/Stiffness Analysis	●					●			●		●	●	●	●	●
Laminate Panel Design				●		●	●	●		●					
Honeycomb Panel Design			●		●										
Panel Buckling		●		●	●	●	●			●					
Modal Analysis				●											
Thermal Analysis	●					●			●	●	●				
Input															
Material Properties	●	●	●	●	●	●	●	●	●	●	●	●	●	●	●
Laminate Layup	●			●		●			●	●	●	●	●	●	●
Panel Dimensions		●	●	●	●			●				●			
Stiffener Data							●								
Cutout Data							●	●							
Ply Thermal Properties	●							●		●	●				
Loading															
Panel In-Plane Loads	●	●	●	●		●	●	●	●	●	●	●	●		
Panel Edge Moments	●			●							●				
Panel Edge Shear	●				●			●		●	●				
Panel Pressure			●										●	●	●
Ply Temperatures	●					●		●		●	●				
Initial Stress / Strain							●							●	●
Output															
Ply Stresses	●					●		●	●		●	●	●		●
Panel Deflection			●	●									●		●
Panel Properties	●								●		●				
Panel Buckling Load		●		●	●	●	●								
Panel Eigenvalue				●											
Panel Edge Moment			●												
Panel Inplane Resultant			●	●				●						●	
Margins of Safety	●					●				●					

Table 3. Structural Joint Analysis Codes

Capabilities \ Codes	A4EI	A4EJ	A4EK	AC-20	BJSFM	BONJO	GRUMFIT	JOINT	JTSDL	JTSTP	SAMCJ	SASCJ	SCAN
Program Capability													
Strength Analysis	●	●	●	●	●	●	●	●	●	●			
Failure Analysis				●	●			●	●	●			
Fatigue Analysis							●						
Thermal Analysis						●		●					
Joint Type													
Bolted		●					●	●			●	●	●
Bonded	●			●	●	●		●	●	●			
Bolted-Bonded			●										
Metallic	●	●	●	●	●		●	●			●	●	
Composite	●	●	●	●	●	●		●	●	●	●	●	●
Lap Joints		●	●	●		●	●	●	●	●	●	●	
Step Joints	●			●				●		●	●	●	
Doublers	●		●										
Input													
Material Properties	●	●	●	●	●		●	●			●	●	●
Adhesive Properties	●	●	●	●		●		●	●	●			
Laminate Material Properties				●	●	●			●	●	●	●	●
Adhesive Thermal Properties													
Fastener Properties		●	●		●		●				●	●	
Thermal Properties						●		●					
Loading													
Axial Loads	●	●	●	●	●	●	●	●	●	●	●	●	●
Shear Loads				●									
Output													
Adhesive Shear Stress	●	●	●	●		●		●	●	●			
Adherend Stresses	●	●	●	●				●	●	●			
Fastener Stresses		●	●				●	●			●	●	
Laminate Dispacement					●		●						●
Laminate Stresses					●	●	●				●	●	●
Laminate Properties					●								
Ply Failure					●						●	●	●
Margins of Safety								●					

Table 4. Optimization Codes

Capabilities \ Codes	AC-120	AC-31	AC-32	AC-33	ADDRESS	ASOP III	ASTROS	OPTCOMP	OPTIM III
Program Capability									
Laminate Strength/Stiffness Analysis	●	●	●	●					
Minimum Weight Design	●	●	●	●	●	●	●	●	●
Laminate Stability	●			●				●	●
Finite Element Analysis					●	●	●	●	●
Modal Analysis					●		●		
Damage Analysis					●				
Flutter Analysis							●		
Element Library									
Truss					●	●	●	●	●
Beam					●	●	●		●
Membrane					●	●	●	●	●
Shear Panel					●	●	●	●	●
Solids							●		
Input									
Material Properties	●	●	●	●	●	●	●	●	●
Cutout Data									
Ply Thermal Properties									
Loading									
Panel In-Plane Loads	●	●	●	●					
Panel Edge Shear	●	●	●						
Ply Temperatures	●						●		
Initial Stress / Strain									
Body Loads							●		
Pressure Loads							●		
Output									
Ply Stresses	●	●	●				●	●	
Ply Strains		●	●				●	●	
Structural Weight	●	●	●			●	●	●	●
Ply Orientation	●	●		●				●	●
Ply Thickness	●	●	●	●					
Panel Stiffness	●	●	●						
Element Stresses					●	●	●	●	●
Element Strains						●	●		
Element Sizes					●	●	●	●	●
Nodal Displacements					●	●	●	●	●
Nodal Forces					●	●	●	●	●
Margins of Safety									

Table 5. Structural Integrity Codes

Capabilities \ Codes	APES	CRACKS IV	CRKGRO	DAMAGE	DAMGRO	LUGRO	MMCLIFE	MODGRO
Program Capability								
Life Prediction			●	●	●	●	●	●
Crack Growth Prediction		●	●	●	●	●	●	●
Failure Prediction					●	●	●	●
Finite Element Analysis	●							
Mission Analysis							●	
Thermal Analysis				●				
Input								
Material Properties	●	●	●	●	●	●	●	●
Crack Growth Model		●	●	●	●	●		●
Crack Growth Rate Parameters		●	●	●	●	●	●	●
Initial Crack Geometry	●	●	●	●	●	●	●	●
Stress Intensity Relationship		●	●	●	●	●		●
Retardation Effect		●	●	●	●			●
Acceleration Effect			●	●				
Nodal Coordinates	●							
Element Connectivity	●							
Thermal Profile				●				
Loading								
Constant Amplitude Cycle		●	●	●	●	●	●	●
Random Spectrum					●			
Flight Spectrum	●	●	●	●	●	●		●
Mission Sequence		●	●	●	●	●		
Panel In-Plane Loads	●					●		
Panel Edge Moments	●							
Panel Edge Shear	●							
Panel Pressure	●							
Ply Temperatures	●							
Output								
Current Crack Size		●	●	●	●	●	●	●
Crack Growth Rate		●	●	●		●		●
Cumulative Cycles		●			●	●		●
Cumulative Flights			●	●	●	●		●
Current Stress Intensity Factor	●	●	●	●	●	●	●	●
Crack Initation Prediction					●		●	
Lamina Failure							●	
Ply Stresses							●	

REFERENCES

1. Pilkey, W. D., Saczalski, K. J., and Sheaffer, H. G., eds., "Structural Mechanics Computer Programs - Surveys, Assessments and Availability," University Press of Virginia, Charlottesville, Virginia, 1974.

2. Perrone, N., and Pilkey, W. D., eds., "Structural Mechanics Software Series," Vol. I, University Press of Virginia, Charlottesville, Virginia, 1977.

3. Perrone, N., and Pilkey, W. D., eds., "Structural Mechanics Software Series," Vol. II, University Press of Virginia, Charlottesville, Virginia, 1978.

4. Perrone, N., and Pilkey, W. D., eds., "Structural Mechanics Software Series," Vol. III, University Press of Virginia, Charlottesville, Virginia, 1980.

5. Perrone, N., and Pilkey, W. D., eds., "Structural Mechanics Software Series," Vol. IV, University Press of Virginia, Charlottesville, Virginia, 1982.

6. Perrone, N., and Pilkey, W. D., eds., "Structural Mechanics Software Series," Vol. V, University Press of Virginia, Charlottesville, Virginia, 1984.